How to make a Virtual Twin of a complex slender structure like a high voltage transmission tower?

T. Tryland

SINTEF Manufacturing, Raufoss, Norway

1 Background

Today it is common to hear discussions related to "industry 4.0", "big data" and the "internet-of-things". These are broad concepts that lack specific details of how they may arise in practice. However, the digital twin has emerged as a potentially transformative idea for engineers engaged in modelling and simulation [1]. A digital twin is a realistic representation that is made with ambition to improve the real product. Remember that the nature always find the lowest energy mode, and the idea here was to make a numerical model that was sufficiently accurate to predict this. The high voltage transmission tower is a slender structure meaning that elements under compression are long compared with their cross-sections. Profiles with larger and thinner walls may therefore contribute to weight saving. It is crucial to find a geometry that contribute with sufficient stiffness to keep the material in position and thereby utilize the material strength instead of allowing it to bend away. The tower is a complex structure, and it is beneficial to find the element that limits the capacity to be able to either strengthen this component or instrument it to measure what is happening during the test. Figure 1 illustrates the tower prior to testing as well as after collapse in the final test. The idea with a sufficiently accurate numerical model is to simulate prior to testing what is most likely to happen in between these two photographs. It is often a challenge to get good measurements and pictures of the critical detail when testing a large assembly. Numerical simulations can be useful to identify the critical elements, plan the best test arrangements and understand what was really tested. In addition, a representative numerical model is a good base to evaluate ideas and find solutions that improve the product.

Fig.1: Pictures of the tower prior to testing as well as after collapse in the final test.

The obvious questions is how to combine models and data to create a virtual prediction tool? There is a long history with measured data to adjust finite element models representing the geometry and material properties of the system. It seems to be a good starting point to represent the initial geometry with correct stiffness and challenge the element formulations to maintain realistic stiffness even when the elements are severely deformed. Figure 2 illustrates how increased computational power has resulted in models that are more detailed. Starting with very coarse models in 1986 there is no doubt that more elements was required to represent the geometry [2]. Today the car models have become significantly more detailed than the full vehicle example illustrated here with sufficiently many nodes to represent the overall geometry. It is therefore relevant to evaluate the need for even more shell elements when their dimensions get closer to their thickness [3, 4, 5]. A proper discretisation is crucial when building a finite element model, and most metallic parts are represented as continuums although we know about the elementary particles. In addition, even relatively large parts like corners and holes with radius close to the element size may be left out in a finite element model. Many CAD geometries includes details with negligible influence on the stiffness compared with the negative effect of poor mesh quality when these details are not neglected. The deformable barrier used in the 64 km/h frontal offset test is an example where the cell structure was too small to be modelled with a reasonable number of shell elements, and the first numerical model was made with solid elements to represent the honeycomb blocks. This required development of a special element formulation to handle the severe deformation of the solid elements together with a special material model to handle the extreme anisotropy [6]. It is herein important to notice the amount of work, the uncertainties with the test specimens and the test procedure to get a proper representation of the honeycomb as a material. It was therefore suggested introduce some scaling of the cell dimensions to represent the barrier geometry with shell elements that were able to capture the deformation mode with local and global buckling of the honeycomb structure together with a simple model to represent the material in the 0.076 mm thick aluminium foil [7].

Fig.2: Increased computational power has resulted in FEM models with more elements and details.

2 Do the right things as simple as possible

It is important to “do the right things as simple as possible” and thereby avoid over-engineering which is a likely result of “do everything right”. It is a challenge to measure ductility when the material is cut from thin-walled sections. It has to be kept in place to evaluate the behaviour at severe deformation, and a complex S-shaped test specimen was found useful at the university [8]. Figure 3 illustrates technology and process utilization to evaluate a specimen holder that fits inside a uniaxial tensile test machine and a simple test specimen that has limited size outside the critical shear region [9].

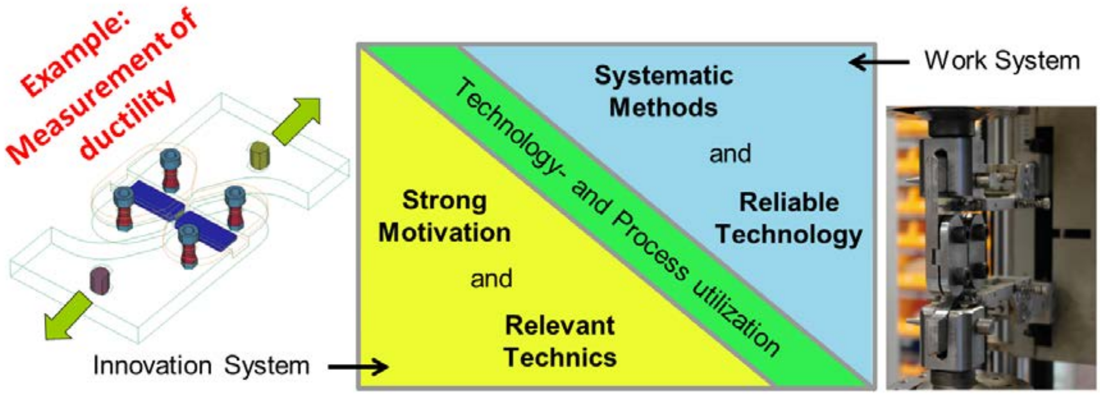

Fig.3: Example to illustrate a successful transfer from the innovation system into the work system.

The idea with figure 3 is to illustrate an approach that supports the idea of detecting everything that may be important, take into account everything that has significant effect and leave out minor details. As an example, the numerical model at left hand side in this figure was used to evaluate the effect of some gap between the parts because of the tolerances when machining the specimen holder and the test specimens. A distance in the range 0 - 0.04 mm seems acceptable even dough it has some effect on the first part of the response curve. However, the specimen holder should not be clamped inside the test machine. The loading has to be applied through a bolt at each end the secure nearly free rotation when the deformation concentrates like a shear zone. Remember that the last point measured corresponds with the state where the elastic energy in the test arrangement is equal to the energy required to tear of the test specimen. Inverse modelling to define the material properties confirm that it is a good starting point to represent the initial geometry with correct stiffness and challenge the element formulations to maintain realistic stiffness even when they are severely deformed.

The stiffness is the critical element for slender structures like a high voltage transmission tower. Therefore, the idea here is to use a combination of element type and element mesh that predict the correct deformation mode and use the simplest possible material model to handle phenomena related to the material. Remember that a digital twin is a realistic representation that is made with ambition to improve the real product. The obvious questions is how to combine models and data to create a virtual prediction tool? Figure 4 shows the deformation modes for different lengths of a sensitive tube geometry under axial compression. The tests were performed with five replicates that show the same deformation mode. Note that the shortest variant defined by L = 2D = 60T shows four concertina rings when it is deformed to half of the initial length [10]. The longer variant L = 4D = 120T demonstrates the sensitive geometry as a transition into 2-lobe buckling after 5 – 6 concertina rings. The long variant defined by L = 6D = 180T shows global buckling with interaction from local buckling, while the longest demonstrates global buckling and lower capacity than the shorter ones. The ambition to predict correct stiffness requires either shell elements with aspect ratio above four to predict the transition from local to global buckling at L = 6D. Alternatively, solid elements are useful to capture the deformation modes with the details shown in figure 4 [11]. It is worth to notice that each solid element should be like a cube geometrically to handle the stiffness properly, and third order elements may be required to handle severe deformation as the element shape becomes different from the initial cube. However, for practical engineers **ELFORM = ÷2** may be useful since it improves the initial stiffness for solid elements with aspect ratio in the range 1 – 4 between solids like cubes and sufficiently large shell elements [12, 13].

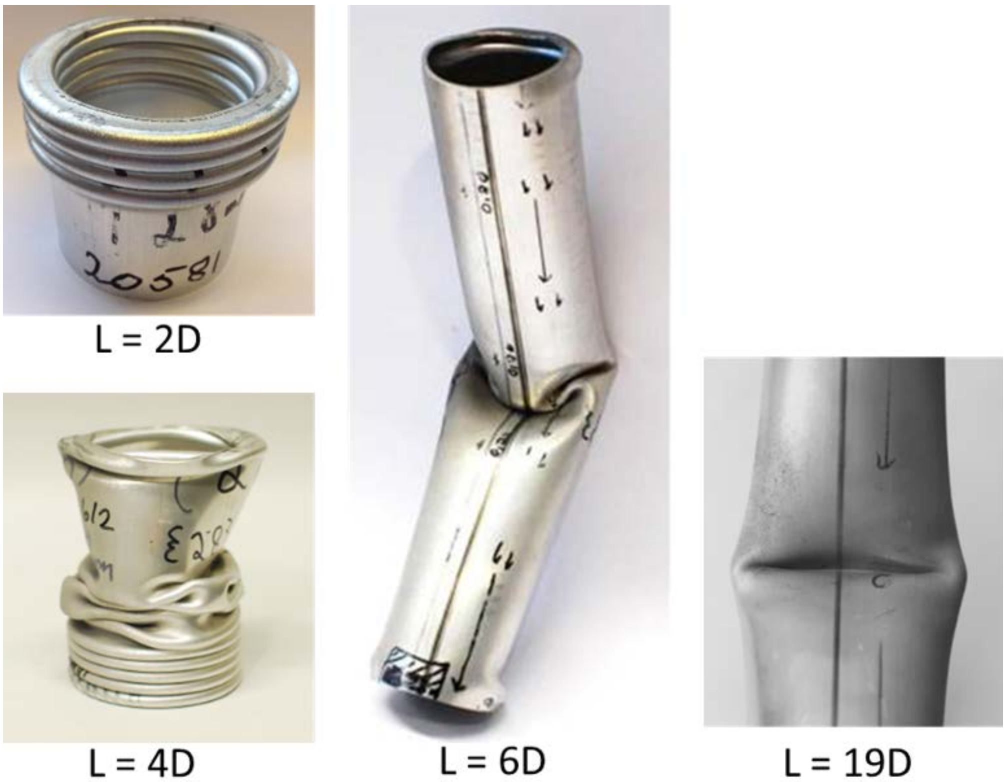

Fig.4: Deformation mode for different lengths of a sensitive tube geometry under axial compression.

The idea behind effective learning based on FEM is elements that handle the deformation mode at the relevant level of detailing and a material model that handle the properties in a metallic material that have a metallurgical explanation. It is likely that a split like this will improve the learning effect as it reduces the amount of curve fitting of parameters without a direct link to physical phenomena. The individual focus in the learning process is important to obtain maximum benefit from the four stages in figure 5 [14]. It seems like realistic expectations of what is the most likely deformation mode is useful to be able to build a model with realistic stiffness. As an example figure 5 illustrates how experience with an A4 paper model may support the training required to detect the modelling mistake in a new situation based on proper understanding of a uniaxial tensile test. Avoiding misleading results is crucial for effective learning based on FEM. The modelling mistake that should be detected in this example is the numerical connection between the nodes around the hole that adds sufficiently stiffness to avoid tear-out at the ends. It is worth to remember that the lowest mode is always detected in the real world independent of whether this mode is found in the numerical model, and the costs may be high when the mistake is detected too late. It may therefore be an idea to build the digital twin of a high voltage transmission tower using only elements that are visible in the pre- and postprocessor, and use common nodes to connect the parts. This means to avoid tied contacts, springs, beams and other numerical features where its contribution to added stiffness is not visible on the computer screen. These remedies may be harmful in itself, but there is a risk for mistakes after some changes of the model, and it may be harder to find them when they are not visualised. However, it is important to balance the resources spent to make correct models and the benefit it brings. The goal is to build a digital twin defined as a realistic representation with ambition to improve the real product.

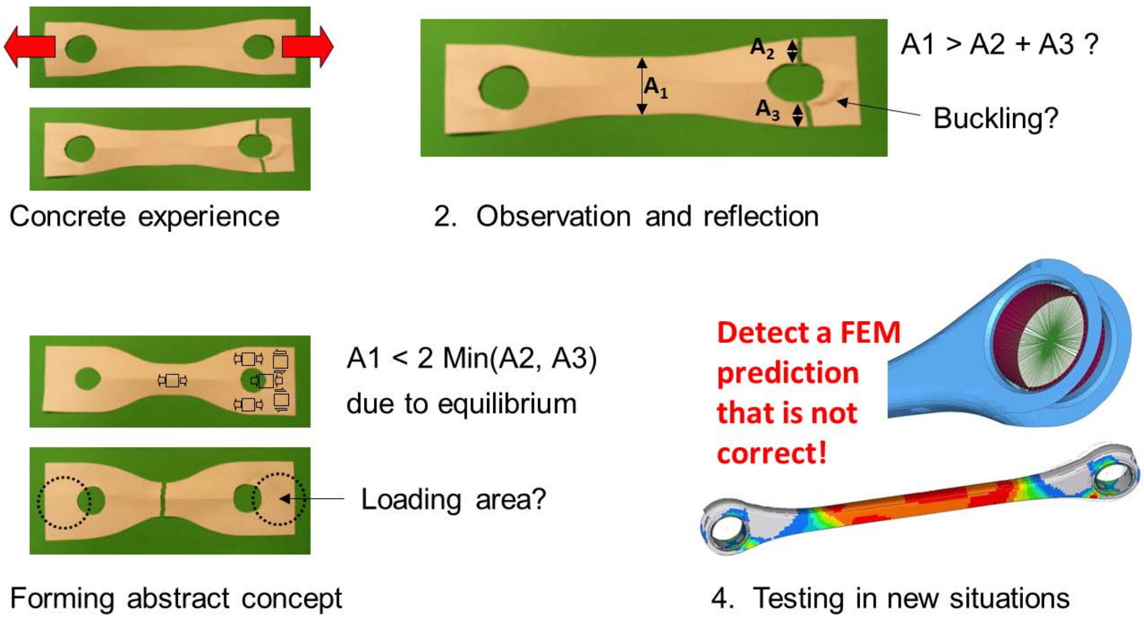

Fig.5: Effective learning to obtain realistic expectations before making a new FEM model [11].

3 The stiffness is the critical element for slender structures

The objective of this study was to investigate whether the same idea can be used to predict instability and the deformation mode when a high voltage transmission tower is loaded until collapse. The tower is made with a relatively large profile in the four corners, see figure 1, and the fundament to the ground has to take both tension and compression. The tower is a slender structure meaning that the elements are long compared with the cross-section dimensions. Aluminium has about one third of the elastic modulus compared with steel, and the idea was to use larger cross-sections to compensate lower material stiffness and save weight due to one third of steel density. The extrusion technology is flexible, and the corner profile was planned with a closed part to secure torsional stiffness and free flanges for simple connections. Figure 6 shows how the stiffness is maintained as far as possible by placing the bolted connections as far out from the profile neutral axis as possible. Note that the interaction between local and global buckling determine the dimensions of a thin-walled large profile with minimum weight, and it was therefore important to use shell elements larger than four times their thickness. However, some smaller shell elements was accepted since each bolt connecting three parts was represented by two reasonable sized volume elements placed in between the parts to connect and with common nodes with these. Figure 6 illustrates the balance between keeping 30 mm as element size to capture the stiffness of the 8 mm thick free flange and the idea of a reasonable stress level when representing a M16 bolt with common nodes. The results of smaller shell elements in this area is somewhat too low local flange stiffness around the bolts and somewhat higher local stresses. The thickness of the plates on each side of the free flange was 16 mm, and modelling one M16 bolt with two 15x15x12 mm solid elements to connect these parts is a rough representation. The cross-section area of the bolt is relatively close, but this representation does not capture the edge contact since is it has an unphysical connection at the opposite side where the bolt may move away from the edge. However, modelling the bolted connections like this seems to be more representative with respect to stiffness than tied contacts between some nodes. In addition, the size of the volume elements representing the bolts is kept reasonable related to the bolt dimensions and it make sense to adjust the stiffness to correspond with a simple test. Figure 7 shows a simple test arrangement to measure the stiffness of a simple bolted connection under tension and compression. The idea was to use a closed cross-section to improve the stiffness, and cut the profile ends with an angle about 45 degrees to get access to the bolt. Note that this solution introduce eccentricity, but it is likely that this is harmful as most of the bolted connection in the suspension tower is fixed to larger profiles with torsional stiffness to handle this.

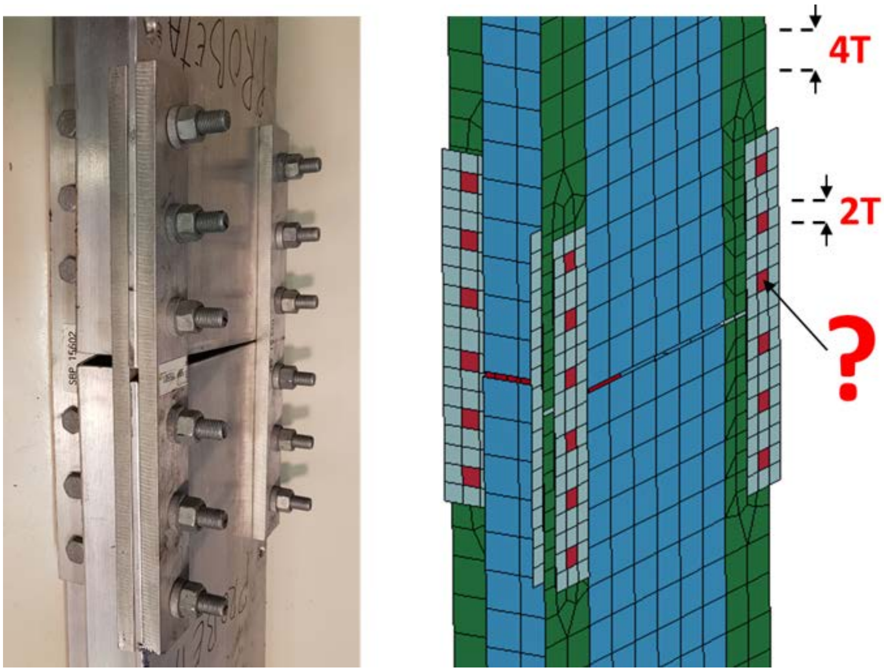

Fig.6: Bolted connection after the final test and model of this connection with solid elements as bolts.

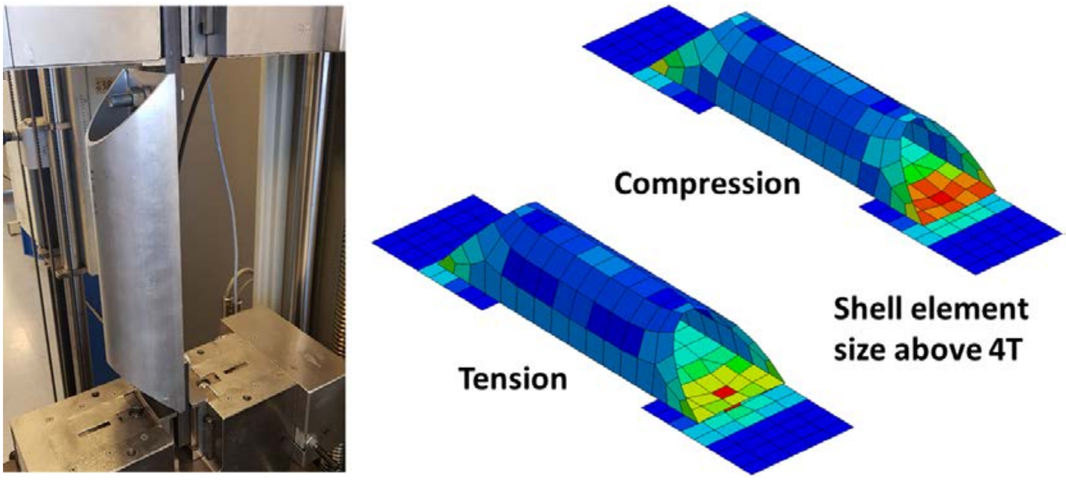

Fig.7: A simple test to measure the stiffness of a bolted connection under tension and compression.

The measured force displacement curves from the test with a simple bolted connection is shown in figure 8. Note that the tests were performed with no torque during the first 5 kN, but then 150 Nm torque was applied to the bolts for about half of the tests. It is clear that the stiffness increases significantly for the tests with pretension of the bolts, while the ultimate capacity is somewhat higher. The tests with tension show tear-out of the bolt at the weakest end, while the tests with compression show plastic deformation in a local area in front of the bolt. The stiffness in tension of one bolted connection seems to be about 40 kN/mm with one M16 bolt with 150 Nm torque to connect an 8 mm thick aluminium plate and a 20 mm thick steel plate. The stiffness in compression seems to be about 60 kN/mm. However, the stiffness without pretension of the bolt seems to be about half of the values above. Note that each M16 bolt in figure 7 is represented by one 15x15x14 mm solid element to connect the two parts, and the elastic modulus of the material representing the bolt is adjusted to correspond with the measured stiffness.

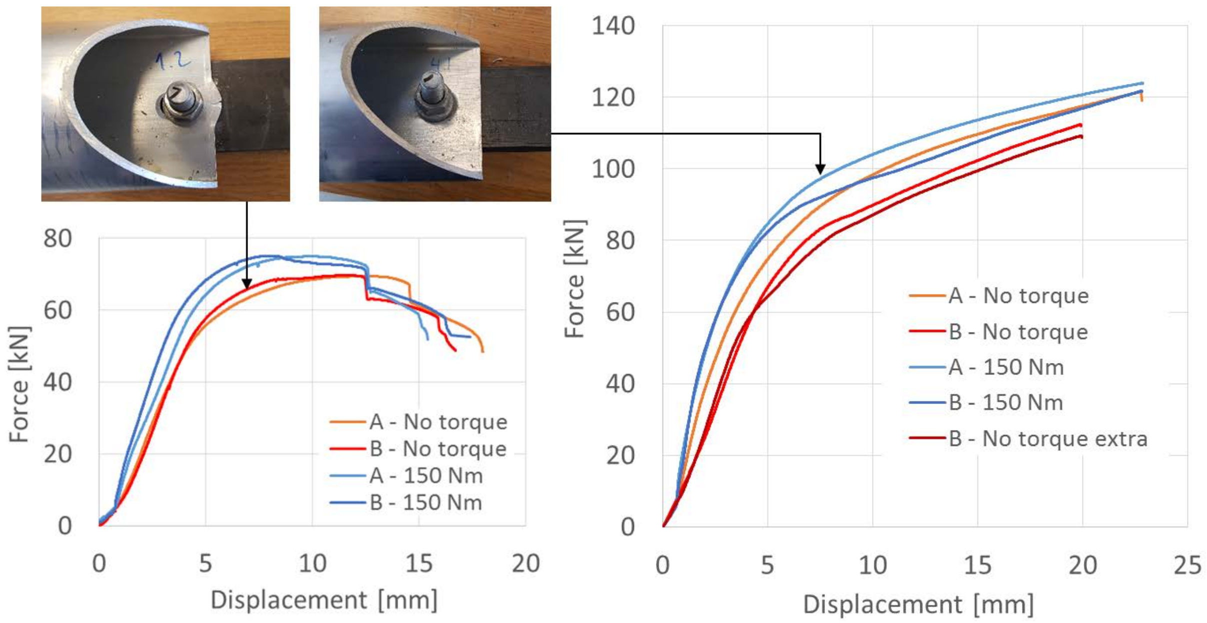

Fig.8: Results from a simple test to measure the stiffness of a bolted connection at both ends.

Figure 9 shows detailed modelling of the test to measure the stiffness of a bolted connection. The geometry is represented by volume elements like cubes to handle the stiffness correct, and realistic material properties can be determined from a uniaxial tensile test or a shear test [9]. The results show local deformation in front of the bolt under compression, while tear-out is predicted when tension is applied to the connection. A complete transmission tower with this element size is questionable.

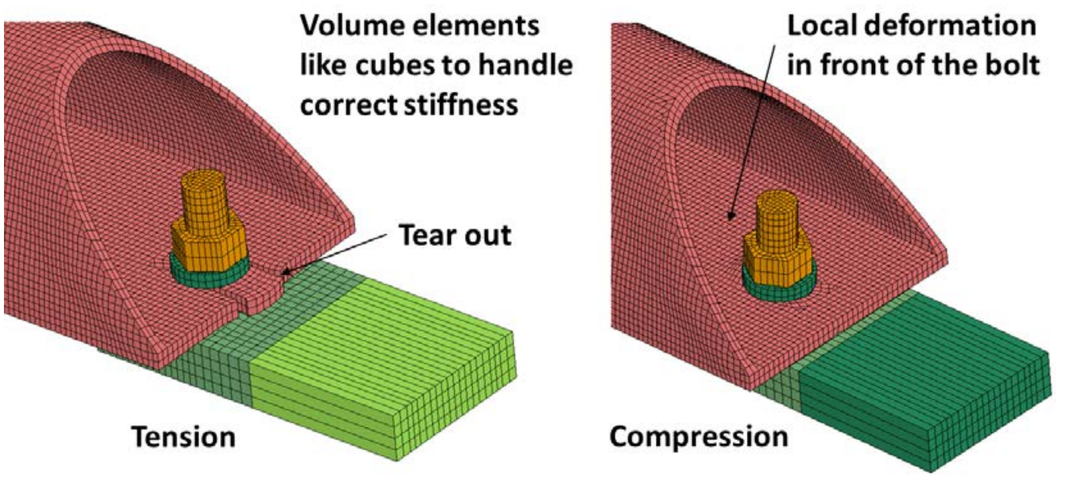

Fig.9: Detailed modelling of a simple test to measure the stiffness of a bolted connection.

4 Effect of eccentricity for long members under compression

It is worth to notice that the profile shown is figure 7 is short to evaluate the stiffness of the bolted connection. This profile is used in the transmission tower as well, but as longer members to connect two of the profile in the corners to form one tower leg as shown in figure 10. The transmission lines may add extra weight as ice, and this may fall down on one side along the lines. The result may be a severe load case with bending and twisting of the transmission tower. The profile shown in figure 7 has an important function here to carry the shear force down the tower leg. The loading direction determine whether the elements become under compression as shown in figure 10 or under tension with the opposite loading direction. The elements closer to the ground are longer, and it may be beneficial to adjust the orientation to reduce the loading of the longest ones. The result may be as illustrated in this figure where three elements are all very close to their maximum at this load level. Remember that the ultimate goal for a weight-optimised structure is that all elements meet their capacity at the same critical load. If not there is potential to remove some material from those that are not critical. However, this is a complex operation as there is several load lases. In addition comes focus on cost, some extra weight may be tolerated, and this has to be balanced with the cost saving related to standardization of the elements required to build the transmission tower.

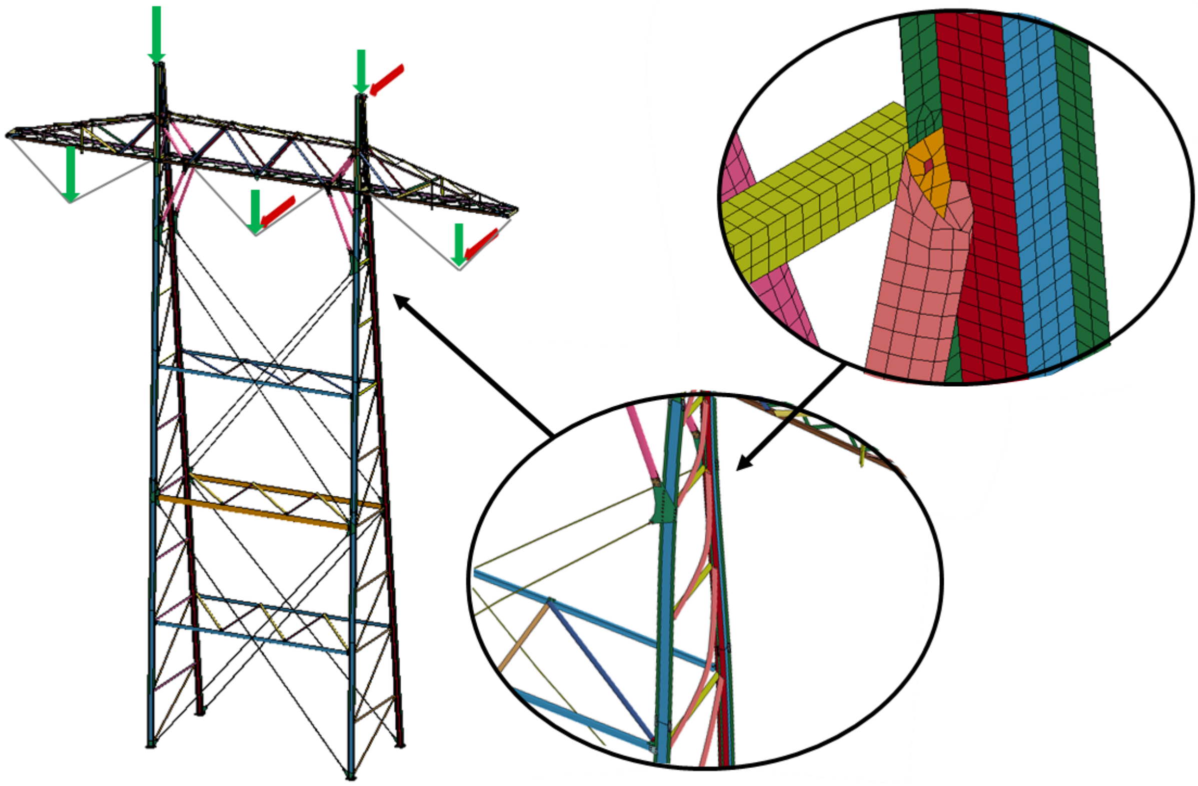

Fig.10: The load case with varying ice loads along the lines shows several critical elements.

The simple test shown in figure 7 was designed to measure the stiffness of a bolted connection, and it may be discussed whether this test arrangement means any effect of eccentric loading when the specimen is this short. Figure 11 illustrates the effect of the distance 18 mm between the mass centres of the profile cross-section and the loading plate for the longest section of this profile used in the transmission tower. The simulation indicates that the profile is nearly straight at one third of the ultimate capacity, and the highest stress component is compression. Some bending is predicted at two third of the ultimate capacity, and the highest stress values in tension and compression have similar absolute values. The profile at this length reaches the ultimate capacity when the bending starts to increase uncontrolled. It is clearly visible with an amplitude much larger than the initial 18 mm eccentricity. The highest stress value is 164 MPa in tension. Note that this value is well inside the elastic region of the material, and the ultimate capacity of the transmission tower may be predicted with an elastic material model. There is some components in the tower like the shortest lengths where the stresses are somewhat higher, but looking at figure 8 indicates that the bolted connection behaves close to linear at 60 kN and it is possible to use two bolts as well.

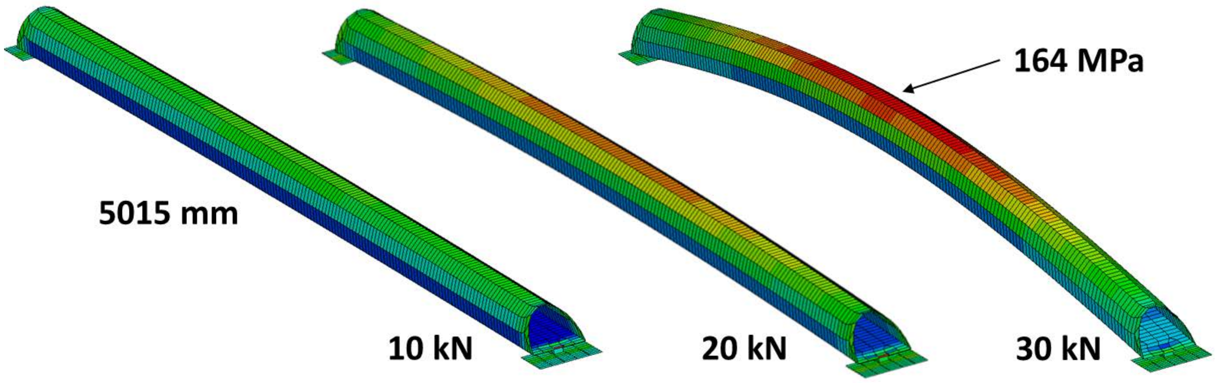

Fig.11: Simulation of a simple test as shown in figure 7 but with the longest section of this profile.

5 Hand calculations may be seen as a virtual twin

Hand calculations following for example British Standard 8118 may be the preferred alternative for proper dimensioning of structural products [15]. Design formulas like this may be seen as a virtual twin as it has potential to improve the product. The obvious questions is how to combine models and data to create a virtual prediction tool? Remember that the selection of parameters involved in the design formulas is based on in depth understanding of the physical phenomena to describe, and how these parameters should be combined is often a result of conscientious entities. In addition comes physical tests to define the constants in the equations as well as secure that the predicted capacity is on the safe side compared with experimental values. The design codes develop over years as more tests results become available, but it is important to balance the need for more accurate predictions with the increased risk for mistakes when the formulas get more complex to achieve this. Eurocode 9 is one example where experiments in some areas have resulted in more complex equations [16]. The result may be more accurate predictions, but it may be more challenging to imagine the physics in these extra numbers as they may look like curve fitting. In addition, separate formulas to adjust the parameters involved may remove some of the motivation to gain experience of how well the parameters work in similar situations, see figure 5. Remember that the difference between knowledge and understanding is a barrier to overcome before the learning process can start. Therefore, with more complex design formulas, it is likely that personnel doing dimensioning will need more time to reach the level "automatic use" where the most beneficial solution is seen more or less immediately, see figure 12.

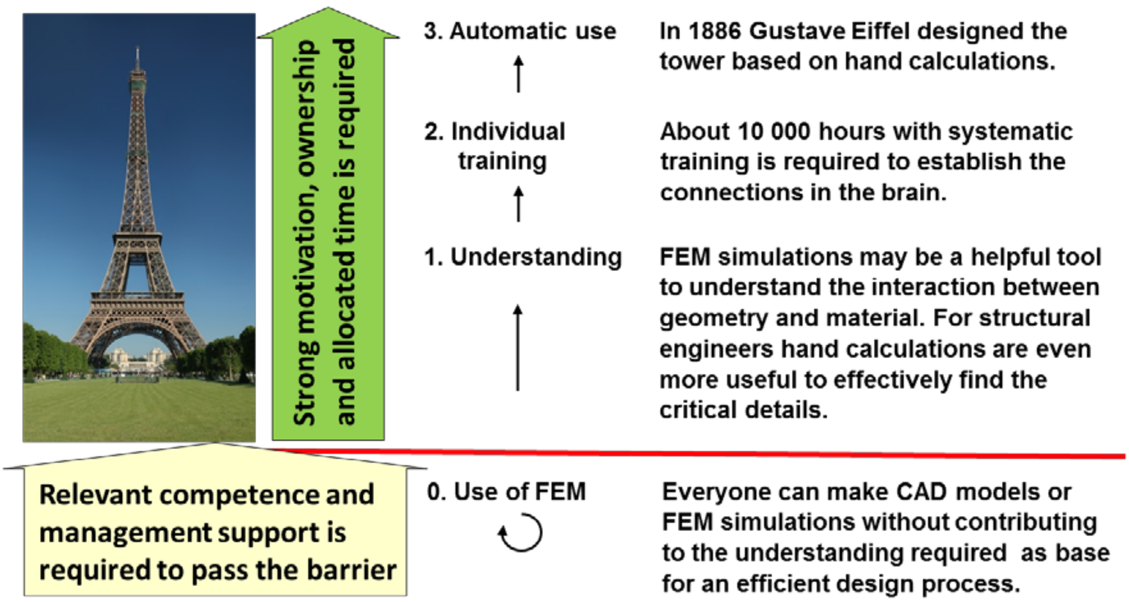

Fig.12: Learning process defined with a barrier before 10 000 hours to reach the level automatic use.

Experienced engineers in their sixties may be able to define a competitive solution geometrically based on dimensioning by hand calculations before it is brought to the drawing table. However a study that compares the competence in physics when Norwegian students entering the universities shows close to two years decrease of maturity during twenty years from 1995 to 2015 [17]. Many students avoid structural engineering as it is challenging courses, and it is likely that the situation may be similar in other European countries. The companies cannot expect the same level of understanding and patience required to rule hand calculations from the younger colleagues that are more impressed by nice colours on the computer screen. It is therefore important to point out a modelling approach that contribute to finite element simulations as a predictive tool that can support improvements of the real product. One example here is buckling of members with varying cross-sections. For hand calculations, this means complex analytical expressions, while a finite element model with realistic stiffness seems useful to understand what the most critical issue is and find solutions that improve the capacity.

Figure 13 illustrates one example where the load case representing maximum wind challenges the out-of-plane stiffness of the connection plate. The idea with this design was to avoid eccentric loading, but it means columns with varying stiffness. Note that the question mark at right hand side point out two weak areas where the bending stiffness is significantly lower than the rest of the thin-walled circular columns. The connection plate has some support in the mid area, but the question is how far from this centre it is ok to have locally reduced stiffness?

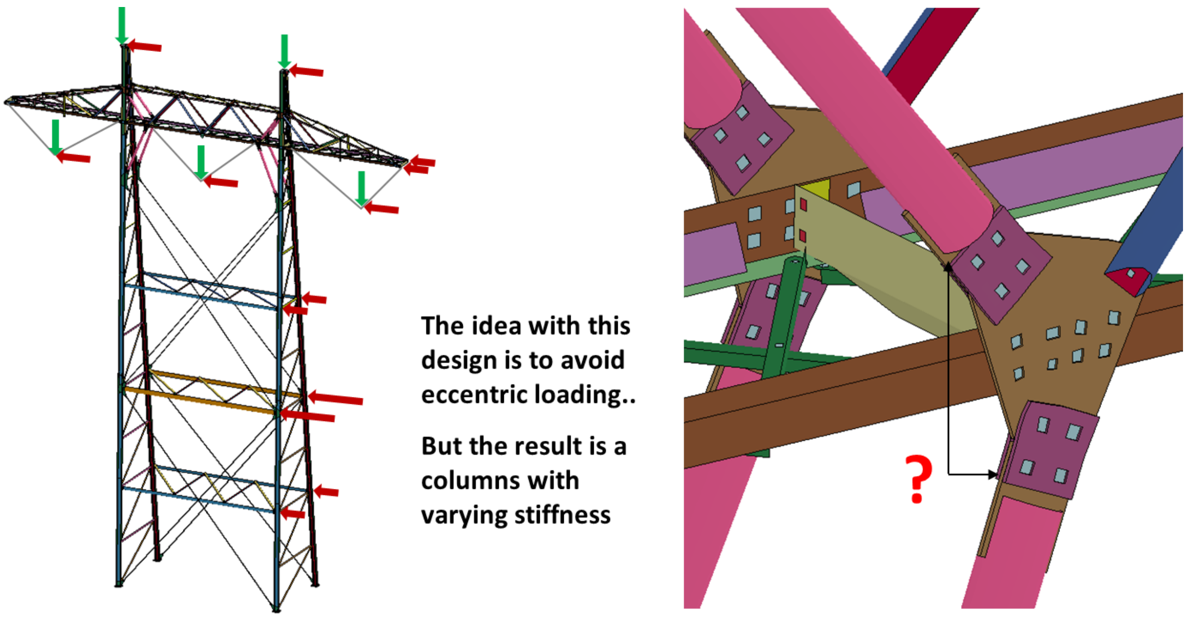

Fig.13: The load case maximum wind challenges the out-of-plane stiffness of a connection plate.

Running a sufficiently detailed finite element model can support hand calculations with realistic assumptions and reasonable simplifications to a level that can be handled by design codes like for example Eurocode. Finite element simulations cannot compensate in depth understanding of structural engineering as base for effective use of hand calculations following the design codes, but it can contribute to learning that is more effective. It is likely that integrated use of hand calculation and FEM to evaluate the assumptions may contribute to the four steps in figure 5: concrete experience, observation and reflection, forming abstract concepts and testing/simulations in new situations like improved versions of the product.

It takes time to build a finite element model with correct stiffness of a complex slender structure like a transmission tower. Therefore, as shown in figure 12, relevant competence and management support it important to secure effective learning towards the state "automatic use". It may be difficult to measure the cost saving related to less resources spent to find the solution that brings most benefit. However, the time saved may be allocated to other activities, and it is likely that this will improve the working environment. It is also important to find the best use CAD and FEM resources. Remember that the value of these models are limited as long as the most beneficial solution is not found. It is often time consuming to make a proper mesh quality based on a full CAD model, and it may be a good idea to limit this to what is relevant for FEM. Then, resources may be saved when the first complete CAD model represents the product with the geometry that will be produced.

6 Results from simulations with a virtual twin of a high voltage transmission tower

Figure 14 shows how a simulation with the finite element model predicts instability of the connection plate and weak area at the end of the circular column. The last prediction before instability and the next one after instability illustrate the physical phenomena that develops quickly as the elastic energy stored in the tower is allocated into the first area that starts to deform uncontrolled. It may therefore be a challenge to get good pictures when testing a large assembly and finite element simulations may be useful to understand what is really tested. The lowest resonant frequency was about 2 Hz, and it seems possible to apply the loading during 2 seconds with smooth transitions to limit the inertia effects. The difference between the predicted capacity when using 2 seconds and 10 seconds was less than one percent, and 2 seconds was used to save computational time. Note that this test was the last one out of ten, and it was run until the maximum capacity was measured like 131 % compared with the value from hand calculations following Eurocode 9 [16]. Note that some of the guy wires comes under compression during the load case illustrated in figure 13, and they were left out to be able to run exactly the same model with both the explicit and the implicit solver in LS-DYNA. It is also worth to notice that it was challenging to run the implicit solver with ideal geometry, and the model was run with geometrical imperfections like 0.5 % of the predicted deformations from the explicit solver at ultimate capacity. The implicit solver could not find the next step in equilibrium at a force level 15 % above the capacity predicted by the explicit solver and somewhat above the measured maximum load. The explicit solver was able to produce a reasonable prediction of the final collapse, see figure 15.

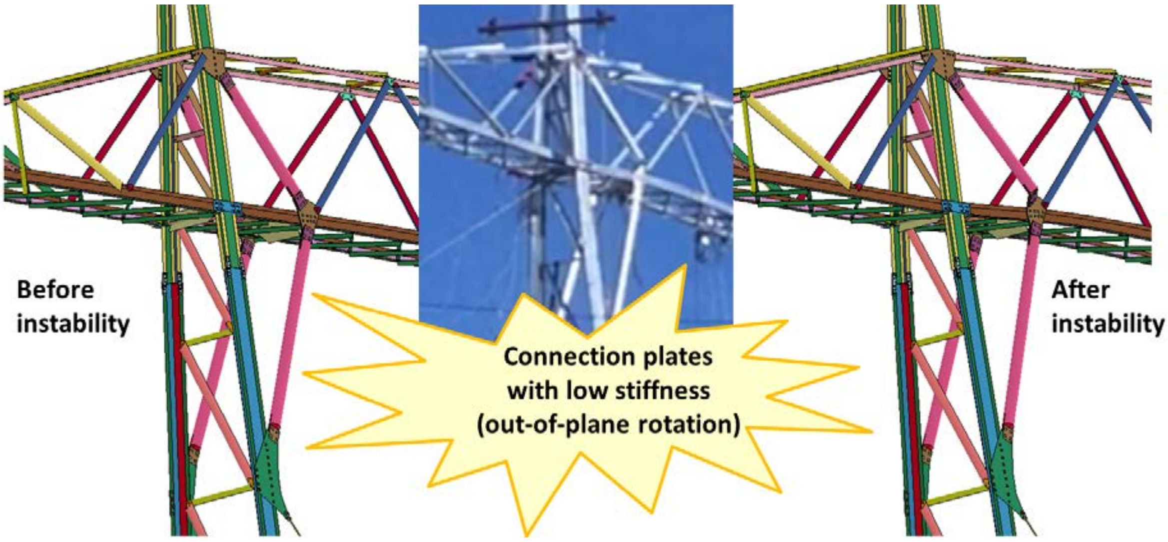

Fig.14: A numerical model with correct stiffness is helpful to understand the lowest mode in the test.

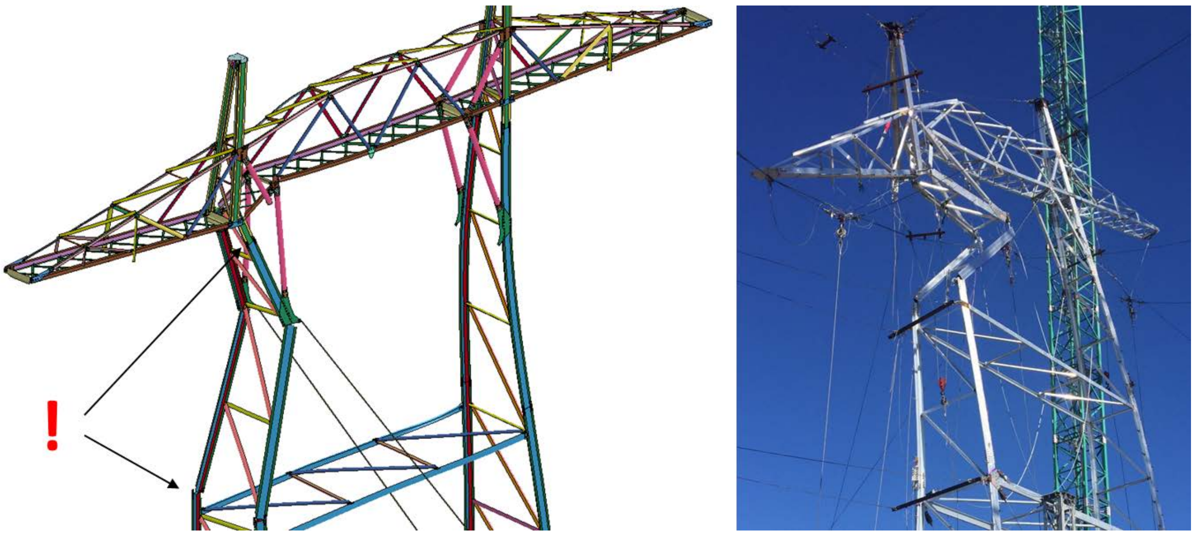

Fig.15: The explicit solver may predict the collapse that take place just after the capacity is meet.

Note that the stiffness limits the load-carrying capacity of a slender structure like this transmission tower, and a linear material model is sufficient to calculate the ultimate capacity. However, the shell elements are large as they have to be at least four times the thickness, and the stress concentrations are underestimated. It is therefore wise to take a closer look at critical areas around bolt holes etc.

7 Possible improvement as a result of a virtual twin with realistic stiffness

Figure 16 indicates the buckling length of the column in this assembly to be about 2630 mm, while simple calculations in accordance with Eurocode 9 indicates that the circular thin-walled cross-section with length 4350 mm can carry this load. It is likely that the weak area with 0.4 % bending stiffness compared with the circular part of the column is placed too far from the supported ends. The idea is therefore to move this weakness to a less harmful location, and a simulation with the model at right hand side in figure 16 indicates 7 % higher capacity even as the eccentricity is increased 12 mm. Note that several parts are replaced by the extended column. It is also worth to notice that the effect of reduced yield stress in the heat effected zone from welding the tube to the plate may become more critical when the stiffness in this area may be improved [18, 19]. The simulations with the explicit solver were therefore repeated with a realistic material model where a reasonable value for the Cockcroft-Latham value reflecting 30 mm shell elements predict the collapse as dramatic as shown in figure 15. It is also indicated that the local stiffness and capacity is increased more than 7 % as a new element becomes the critical one when this improvement is introduced, see figure 17.

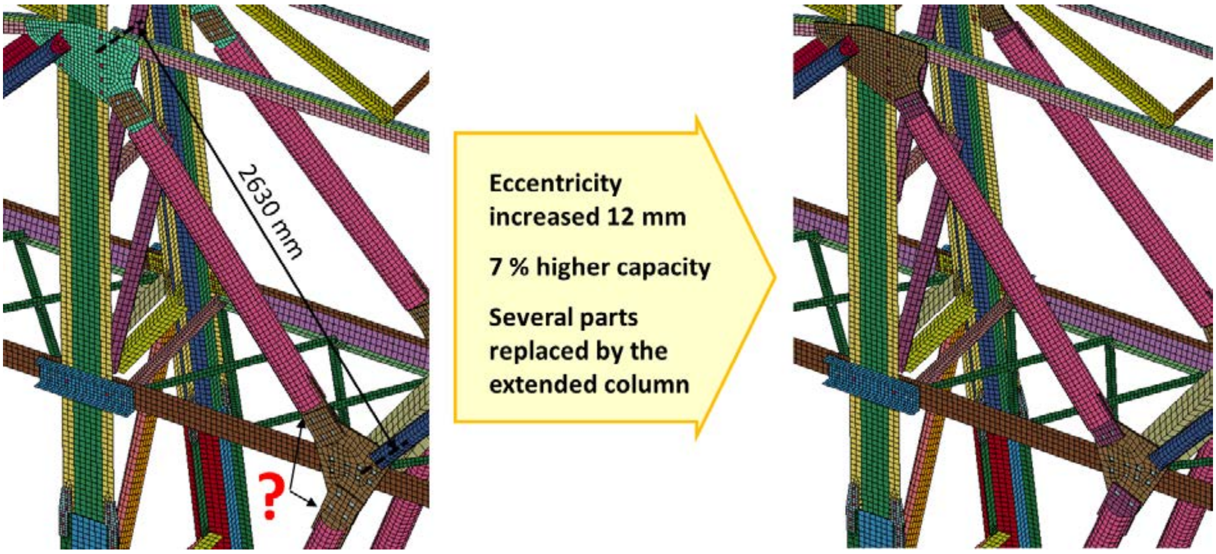

Fig.16: A virtual twin identify what limits the critical element and it is possible to find a better solution.

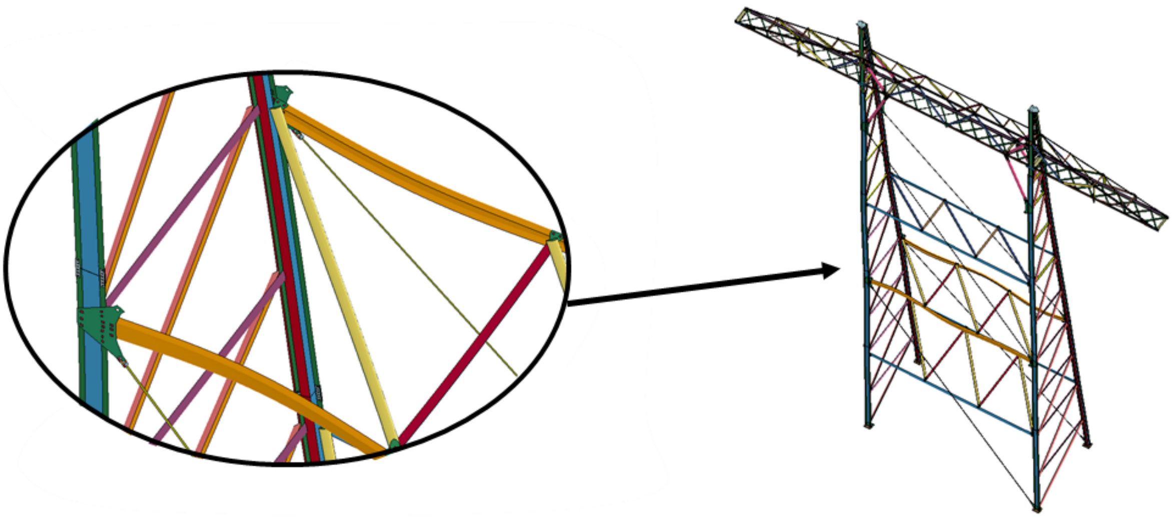

Fig.17: After strengthening the weakest element, a new element becomes the critical one.

8 Summary

It is important to make a virtual twin with realistic stiffness when the ambition is to improve a slender structure like a high voltage transmission tower. This means shell elements that are larger than four times the thickness or solids in case smaller elements are required to predict more geometrical details or stress concentrations. It takes time to make a complex model like this with the mesh quality required, and it is important to find the best use CAD and FEM resources. The idea was to introduce simplifications to be able to run exactly the same LS-DYNA model with the explicit and the implicit solver. Both solvers predicted the correct deformation mode, and the explicit solver was able to predict the final collapse as well. However, the implicit solver predicted 15 % higher ultimate capacity than the explicit one, and the first one was somewhat non-conservative compared with the experimental result.

9 Acknowledgements

The research within this study was supported by grant 245329/O30 from the Research Council of Norway. The support is gratefully acknowledged. Moreover, the support from Statnett SF made this research programme possible.

10 Literature

[1] Wagg, D.: "The Digital Twin for Engineering Applications", NAFEMS Journal engineering analysis and simulations, April issue, 2018.

[2] A free Ford Taurus LS-Dyna model developed 2010-2014 within a founded project of Automotive Simulation Center Stuttgart based on an underlying detailed FE-model from the National Crash Analysis Center at George Washington University, 2001.

[3] Tryland, T.: "A Simple Compression Test to Evaluate Ductility", 9th Nordic LS-DYNA Users Forum, Gothenburg, Sweden, 2008.

[4] Fyllingen, Ø., Hopperstad, O.S., Hansen, A.G., Langseth, M.: "Brick versus shell elements in simulations of aluminium extrusions subjected to axial crushing", 7th European LS-DYNA Conference, Salzburg, 2009.

[5] Haufe, A et al.: "Predictive Fracture Modelling in Crashworthiness: A Discussion of the Limits of Shell-Discretized Structures", 14th LS-DYNA Forum, Bamberg, Germany, 2016.

[6] Kojima, S., Yasuki, T., Mikutsu, S., Takatsudo, T.: "A Study on Yielding Function of Aluminium Honeycomb", 5th European LS-DYNA Users Conference, Birmingham, UK, 2005.

[7] Tryland, T.: "Alternative Model of the Offset Deformable Barrier", 4th LS-DYNA Forum, Bamberg, Germany, 2005.

[8] Achani, D.: "Constitutive models of elastoplasticity and fracture for aluminium alloys under path change", PhD thesis, page 133-135, ISBN 82-471-7903-2, Department of Structural Engineering, NTNU, Trondheim, Norway, 2006.

[9] Tryland, T. and Berstad, T.: "A Simple Shear Test to Evaluate Material Ductility based on Specimens cut from Thin-Walled sections", 11th LS-DYNA Forum, Ulm, Germany, 2012.

[10] Tryland, T.: "Combinations of Meshes and Elements that seems able to Predict the Correct Deformation Mode", 13th LS-DYNA Forum, Bamberg, Germany, 2014.

[11] Tryland, T.: "Tests with a Sensitive Specimen Geometry confirm solid elements when the Aspect Ratio is below four", 14th LS-DYNA Forum, Bamberg, Germany, 2016.

[12] Borrvall, T.:"A heuristic attempt to reduce transverse shear locking in fully integrated hexahedra with poor aspect ratio", 7th European LS-DYNA Conference, Salzburg, 2009.

[13] Tryland, T., Berstad, T.: "Keep the Material Model simple with input from Elements that predict the Correct Deformation Mode", 10th European LS-DYNA Conference, Würzburg, Germany, 2015.

[14] Colb, D A.: "Experimental Learning", Englewood Cliffs: Prentice Hall, 1984.

[15] BS 8118: Structural use of aluminium, Part 1. Code of practice for design, ISBN 0-580-19209-1, Civil Engineering and Building Structures Standards Policy Committee, 1991.

[16] Eurocode 9: Design of aluminium structures, Part 1-1: General structural rules, European Committee for Standardisation, prEN 1999-1-1: 2003.

[17] Grønmo, L.S., Hole, A., Onstad, T.: "Ett skritt fram og et tilbake. TIMSS Advanced 2015. Matematikk og fysikk i videregående skole", Report in Norwegian, University in Oslo, 2016.

[18] Lademo O.-G., Berstad T., Eriksson M., Tryland T., Furu T., Hopperstad O.S., Langseth M.: " A model for process-based crash simulation", Journal of Impact Engineering, Vol 35, March 2007.

[19] Tang, W., Fjær H.G.: "Integrated Forming, Welding and Crash Simulations for an Aluminum Crash Management System", ASME International Design Engineering Technical Conference and Computers and Information in Engineering Conference, Boston, US, 2015.